ELSEVIER
CrossMark
Available online at www.sciencedirect.com
ScienceDirect
Procedia Engineering 99 (2015) 593 – 599
Procedia Engineering
www.elsevier.com/locate/procedia

"APISAT2014", 2014 Asia-Pacific International Symposium on Aerospace Technology, APISAT2014

Influence of High-temperature Air on Three-dimensional Nozzle Structure Design

Bu Junhui*, Liu Huaiyin, Wang ying & Sun Yongtang

China Academy of Aerospace Aerodynamics, 100074, Beijing, China

Abstract

In this paper, the exit diameter of nozzle is Φ 1.4 m, and the Mach number is 4.0; the nozzle is used in high-speed free jet wind tunnel for engine test with high temperature and pressure air and the mixture of heater combustion as working medium; The mixture is high temperature gas, mainly contains CO2 ,N2 and H2O. In the process of nozzle structural design, consider the interaction of radiation heat transfer and convection heat transfer fully; heat transfer problems aimed at the nozzle in complex gas, solid, liquid multiphase flow, form the calculation model of the radiation- convection - steady state – convection. Calculated by engineering and Ansys methods, The results show that, the convection heat transfer has an important effect on the air- surface by the high-temperature air in the entrance and the thorat of nozzle; Convection heat has a great influence on the air-surface temperature distribution of nozzle. It must be applied to the solution of temperature field in the nozzle structure design.
A great deal of experiments show that, the temperature of the nozzle entrance and exit on the air-surface is close to the calculation results. The experimental results show that the nozzle works very well. In this paper, the calculation is described in details. The teat transfer calculation and some typical experimental results of three-dimensional nozzle and the photographs of the nozzle are also presented.

Peer-review under responsibility of Chinese Society of Aeronautics and Astronautics (CSAA)

Keywords: Convection heat transfer;Three-dimensional nozzle; Structure;Flow field.

1. Introduction

Since the arms race tend aerospace, Due to the increased difficulty of optimization of aircraft shape, some

* Corresponding author. Tel.: +86- 18811069098; .
E-mail address: bjhht11@163.com

Peer-review under responsibility of Chinese Society of Aeronautics and Astronautics (CSAA)
doi:10.1016/j.proeng.2014.12.576

countries are committed to the development of high performance engine. The development process of the engine needs a lot of simulation test on the ground, the test technology is complex and costly, so each country attaches importance to the construction and testing technology of test facility. Due to the high temperature, high pressure and the poor working condition in the engine testing platform , design of the structural components become the difficulty of engine testing platform. In this paper, the exit diameter of nozzle is Φ 1.6 m, and the Mach number is 4.0, and the design temperature of this nozzle is 875 k, and the design pressure 2.2 MPa. The nozzle is used in high-speed free jet wind tunnel for engine test with high temperature and pressure air as working medium. It is different from the conventional nozzle design of wind tunnel, it uses high temperature and pressure air and the mixture of combustion as working medium, The mixture is high temperature gas, mainly contains CO_2 and H_2O. Through the establishment of radiation - convection - steady - convection model, combine with ansys to calculate the result, We can get the conclusion that the radiant heat of nozzle inlet section has a major effect on temperature distribution of the nozzle shell, we must be introduced into the temperature field solution during the nozzle structure design. Through the above conclusions, we can obtain relevant evidence and algorithms in the similar situations with nozzle structure design.

2. The structural scheme of the supersonic nozzle

The supersonic nozzle is used in high-speed free jet wind tunnel for engine test with high temperature and pressure air as working medium, the exit diameter of nozzle is Φ 1.6 m, and the Mach number is 4.0, and the design temperature of this nozzle is 875k, the maximum flow of airflow is 230kg/s and the design pressure 2.2MPa, and the work hour is less than 90 seconds. The initial structure scheme of the nozzle uses cooling water for the design, the thickness of shell is 3mm and the material is 0Cr18Ni10Ti, The material has good flexibility and strength and thermal conductivity below 400 ℃, and it is used for the initial nozzle with much density of heat flow. The stainless steel shell has 5mm wide cooling water channel, and it is connected through the half shell. The Main effect of outer shell bear internal pressure, the presser of the inner shell is passed to the outer shell by half shell part. and cooling water only pieces of the inner shell and half of them run. half parts using 0Cr18Ni10Ti materials, the cooling water is only running between the inner shell and half shell, the material of the half shell is 0Cr18Ni10Ti and the initial part of nozzle is shown in Fig.1,2.

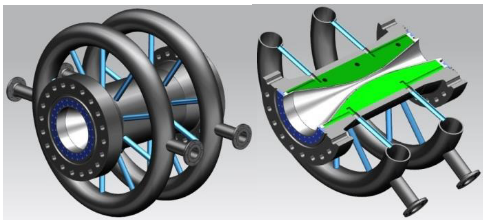

Fig. 1. The first section of the supersonic nozzle

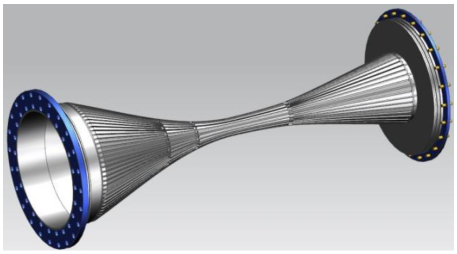

Fig. 2. The inner shell with cooling channel

3. Calculation method

In consideration of mixture with CO2 and H2O that has strong radiation, so the calculation includes two parts, convective heat transfer model and the increased radiation heat transfer model. In the convective heat transfer model calculation process, using a convection-steady-convection heat transfer process model according to the conventional hypersonic wind tunnel nozzle design method, and radiation heat transfer model based on it, and compare the value with each other, so it can be shown that if we need to calculate the initial segment of the nozzle design by radiation heat transfer, in the process of calculation, it is assumed that all the heat could be absorbed by cooling water, so the outer casing is consider of adiabatic wall. The calculation model of heat transfer is shown in Fig.3.

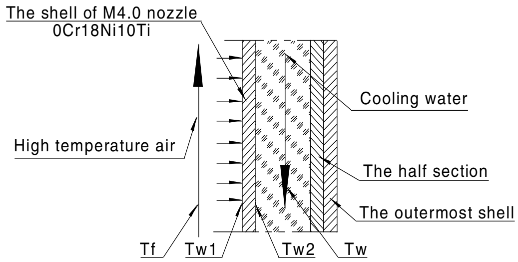

Fig. 3. The picture of heat transfer calculation model

The method of convection-Steady-convection heat transfer calculation is like this, when high temperature fluid flowing through a pipe, forced convection heat transfer occurs in the pipe, as the process of heat transfer is in stable state, if we don't consider the heat loss and the outer jacket of cooling water and air radiation heat transfer, The heat that airflow transfer to the inner shell per unit time by convection heat transfer is not only equal to the heat that transferred to the inner shell of pipe but also equal to the heat that transferred to the outside shell of pipe by convection heat transfer to the cooling fluid, and the last part of heat is equal to the enthalpy increases of cooling water. So we can form a heat balance equation as follows:

$$Q = H_1(t_f - t_{w1})A \tag{1}$$

$$Q = \lambda H_1(t_{w1} - t_{w2})/\delta \tag{2}$$

$$Q = H_2(t_{w2} - t_{H1})A \tag{3}$$

$$Q = c_p m_w(t_H - t_0) \tag{4}$$

In the formula, Q—quantity of heat, W, t_f—the recovery temperature of boundary layer of high temperature air, K, t_{w1}—gas wall temperature, K, t_{w2}—liquid wall temperature, K, t_{H1}—the average temperature of cooling water, K, t_H—the temperature of cooling water outlet，K, t_0—the temperature of cooling water inlet, K, H_1—the convective heat transfer coefficient between high temperature air flow and shell, W/(m^2.K), H_2—the convective heat transfer coefficient between shell and cooling water, W/(m^2.K), λ —the convective heat transfer coefficient of Shell, W/(m.K)；c_p—the specific heat at constant pressure of cooling water at tH1 temperature, J/(kg.K), m_w—the flow rate of cooling water, Kg/s；A—the area of inside shell, m^2, δ —thickness of shell, m. When the parameter of t_f, H_1, H_1, λ ,m_w is known, according to the equation we can get the temperature of gas wall and liquid wall, and Calculate the thermal stress by this. when high temperature fluid forced convection heat transfer occurs in the pipe, calculating the Re at first and judging the flow state, then we can select the formula to calculate. When smooth shell is in the turbulent state, the temperature of gas and shell vary widely, it can choose the Sieder-Tate model to calculate.

Heat transfer coefficient: $\alpha = \frac{\mathrm{Nu}\lambda}{\mathrm{D}}$ (5)

Air flow Reynolds number: $\mathrm{Re} = \frac{\mathrm{vD}}{\upsilon}$ (6)

Prandtl constant: $\mathrm{Pr} = \frac{\mu \mathrm{C_p}}{\lambda}$ (7)

Nusselt number: $Nu = 0.027Re^{0.8}Pr^{1/3}$ (8)

υ— kinematic viscosity, m^2/s, μ—Dynamic viscosity，Pa.s, According to the formula we can do the heat transfer calculation of the initial part nozzle.

The method of radiation-convection-Steady-convection heat transfer process model is shown that the method is mainly to solve the radiation heat transfer of initial segment of the nozzle.

$$\frac{dl(r,s)}{ds} + (a + R_s)\left|(r,s)\right. = an^2 \frac{RT^4}{P} + \frac{R_s}{4P} Q_0^{4P} \left|(r_, sc)5(s_, sc)d8c\right. \tag{9}$$

In the formula, I—radiation intensity, it is the location and orientation s of the function r, r-space solid angle, a-absorption coefficient, n-refractive index, Rs-scattering coefficient, sc-Scattering direction, T-local temperature. Using P1 spherical harmonics method for solving the radiation transfer equation. So it can get the calculation formula of radiation heat flux Qr.

$$Q_r = -\frac{1}{3(a = R_s) - CR_s} G \tag{10}$$

In the formula, G-incident radiation, C-linear anisotropic phase function coefficient.

4. The process of calculation

(a) In the convection - Steady - convection model, the heat transfer coefficient of high temperature gas calculation.

First of all calculating the heat transfer coefficient α by M4.0 state point, that is the design temperature of this nozzle is 875k, the maximum flow of airflow is 230kg/s and the design pressure 2.2MPa. According to the formula, air is working medium and k=1.4:

$$\frac{T^*}{T} = 1 + \frac{k-1}{2} M^2 \tag{11}$$

T*-total temperature, T — static temperature, M-mach number, It can calculate the static temperature，the state point that relative to the speed of sound a can be calculated using the following formula:

$$a = \sqrt{KRT} \tag{12}$$

Assuming that the gas wall temperature is 310 ℃ of the first section, according to the recovery temperature formula to get the value of $t_{w1} = 313℃$, , according to the fluid mechanics schedule we can find out Pr, it conforms to the condition that assumed, and then take the nozzle flow heat transfer coefficient H = 23.8 W/(m^2. K).

(b) The heat transfer coefficient of cooling water

The jackets of cooling water are connecting with each other, according to the high speed flow turbulence model of high speed air flow through a circular tube for calculation, it can take the heat transfer coefficient of cooling water, then using the formula of (1), (2), (3), (4) to found the equations and solve the temperature of inside and outside jackets t_{w2}.

(c) The calculation of shell stress

According to the thermal stress calculation formula to solve nozzle shell stress calculation.

The gas wall: $$\bar{r}_a = \frac{Ea}{2(\gamma)\ln k}\left[1 - \frac{2k^2 \ln k}{k^2 - 1}\right] \tag{13}$$

The side of cooling water: $$\bar{r}_w = \frac{Ea}{2(\gamma)\ln k}\left[1 - \frac{2\ln k}{k^2 - 1}\right] \tag{14}$$

The tangential stress: $\sigma_t = 0.5283P_0\left(\frac{k^2}{k^2-1}\right) - \frac{2P_w}{k^2-1} - \bar{r}_a(T_a - T_w)$ (15)

The axial stress: $\sigma_e = -\frac{P_e}{F} - \bar{r}_a(T_a - T_w)$ (16)

By the fourth strength theory show that the maximum thermal stress of gas wall is 132MPa.

(d) The method of radiation - convection-Steady-convection heat transfer calculation.

According to the radiation heat transfer model, the using Fluent and Ansys software to solve the heat calculation of initial segment nozzle, then we can get the gas wall temperature of inner shell with t_{w1}.

5. Tables and figures

In summary, comparing with gas temperature of the calculated value and actual measured value through the thermocouple, Thermocouples distributed along the nozzle axis, are arranged staggered according to a 45 ° angle, thermocouple mounting structure is shown in Fig.4, the distribution is shown in Fig.5.

The measured values of whether to consider radiation heat transfer are contrasted with the values of calculation, and the result is shown in Table 1 and Table 2. By comparing the measured and calculated values, the value of with radiation heat transfer is well matched with the measured values.

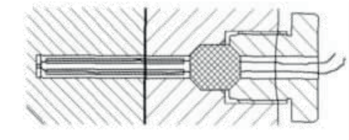

Fig. 4. Thermocouple mounting structure

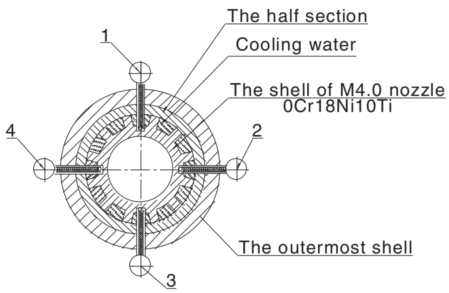

Fig. 5. Thermocouple distributing

Table 1. Measured and calculated values contrast that Ignore radiation heat transfer by thermocouple

NO.	Actual measured value (K)	Calculated value (K)	error%
1	615	611	4.26
2	489	416	-11.48
3	513	420	-14.28
4	527	439	-12.2

Table 2. Measured and calculated values contrast that considered radiation heat transfer by thermocouple

NO.	Actual measured value （K）	Calculated value （K）	error%
1	615	586	4.9
2	489	430	-8.51
3	513	490	4.69
4	527	500	5.4

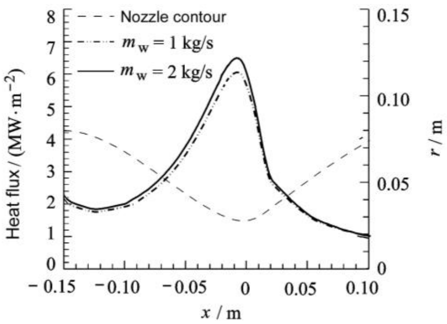

Fig. 6. Contrast heat flux distribution along the axial

Fig.6 is shown that heat flux q distribution of gas temperature, so we can get that distribution of two methods is near the same, only it has some difference near the throat, when considering the case of radiation heat transfer, heat flow density before the throat, is bigger.

This paper calculates the results that weather to consider radiation heat transfer， fig.7 shows the distribution of the gas temperature and the liquid temperature of nozzle in both cases. Fig.8 shows theinfluence of radiation heat transfer that is effect on nozzle temperature, the gas temperature and the liquid temperature is higher in case of radiation heat transfer before the throat, and is more consistent with the actual situation.

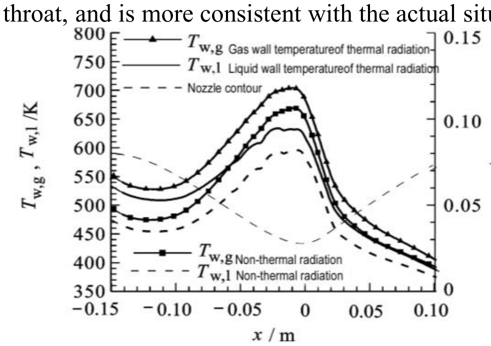

Fig. 7. Gas temperature and liquid temperature distribution

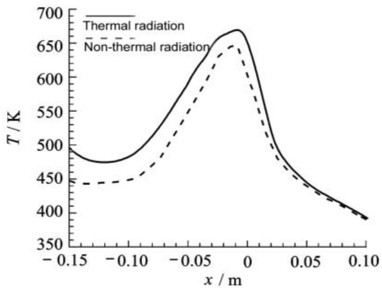

Fig. 8. The temperature Influence on the nozzle of radiation heat transfer

Through the velocity field calibration, direction field calibration and the standard model calibration, we obtained all kinds of test data. The analysis of flow field calibration results are completed in strict accordance with the GJB1179-91" high speed and low speed wind tunnel flow quality standard". We can see M4.0 nozzle by using in Fig.9.

Fig. 9. The picture of M4.0 nozzle in factory

6. Conclusion

- In the supersonic nozzle design, if the working medium containing a mixture of CO2 and H2O and other impurities, it should be consider with the heat exchanger in the inlet nozzle section with radiation heat transfer particularly.
- If the gas temperature is higher and contain strong radiation gas nozzle, the temperature of radiation heat transfer has a major impact, in this case, the radiation of high-temperature can not be ignored and must be introduced into the solution of the temperature field.
- A great deal of experiment data show that the nozzle flow field stable performance is good, fully satisfies the various flow field metrics.

References

[1]. Kegelman J T, Roos F W, Influence of forebody cross-section shape on vortex flowfield structure athigh alpha[R].AIAA-91-3250, 1991.
[2]. Sun Jufang, SUN Bing, Xiang Shenghai, Theoretical calculation of complex nozzles temperature profile and stress profile in solid rocket motor[J]. Journal of Propulsion Technology, 1994, 15(1):23-31.
[3]. GJB 1179-91.High speed and low speed wind tunnel flow quality standard. Science and technology Committee of National Defense, 1992.6.
[4]. Cheng Daxian, Manual of mechanical design.Chemical Industry Press 2002.11.
[5]. WENG Zhongjie, Measurement of the transient tempera-ture on inside wall of small typed rocket nozzle[J].Journal of Propulsion Technology, 1984, 5(1):51-55.
[6]. Liu Zhengchong, Aerodynamic and structural design of high speed and low wind tunnels. National Defense Industry Press 2003.4.